



Yang, L., Yang, H., Yan, S., Ma, Q., & Bihnam, M. (2016). Comparative Study on Water Impact Problem. In *The 26th International Ocean and Polar Engineering Conference, 26 June-2 July, Rhodes, Greece* (Proceedings of Annual International Ocean (Offshore) and Polar Engineering Conference (ISOPE)). International Society of Offshore and Polar Engineers.  
<https://www.onepetro.org/conference-paper/ISOPE-I-16-500>

Publisher's PDF, also known as Version of record

License (if available):  
Unspecified

[Link to publication record in Explore Bristol Research](#)  
PDF-document

This is the final published version of the article (version of record). It first appeared online via ISOPE at <https://www.onepetro.org/conference-paper/ISOPE-I-16-500>. Please refer to any applicable terms of use of the publisher.

## University of Bristol - Explore Bristol Research

### General rights

This document is made available in accordance with publisher policies. Please cite only the published version using the reference above. Full terms of use are available:  
<http://www.bristol.ac.uk/red/research-policy/pure/user-guides/ebr-terms/>

# Comparative Study on Water Impact Problem

Liang Yang, Hao Yang, Shiqiang Yan, Qingwei Ma and Maria Bihnam  
School of Mathematics, Computer Science and Engineering, City University London  
London, UK

## ABSTRACT

This paper presents a comparative numerical study for the water impact problems due to dropping of triangular wedges or ship sections. In the numerical investigation, both the dynamic mesh technique and immersed boundary method adopting fixed Cartesian grids have been adopted in order to conform to the motion of the structure. For the former, a multiple-phase solver with the volume of fluid for identifying the free surface is implemented. In the simulation using this method, both the compressible and incompressible solvers have been considered to explore the role of the compressibility. For the latter, an in-house immersed boundary method, in which a generalized equation is developed to govern the motion of different phases (air, water and solid) and a level-set method is adopted to identify the free surface & body surfaces. Different cases with different dropping speed have been considered in the investigation and the results are compared with the experimental data for the comparative study on the water impact problem.

**KEY WORDS:** Water impact; comparative study; immersed boundary method; CFD

## INTRODUCTION

The water entry is a complex, high-speed and nonlinear fluid-structure interaction problem covering many physical phenomena, such as the air trapping, free surface deformations, spray and jet flows. Significant impulsive pressure and slamming forces associated with the water entry problems lead to considerable hydro-elastic issues and, possibly, a severe damage of the offshore structure. Although this problem has been attracting the awareness of industrial and academic communities, the relevant understanding is still developing, in particular the role of the compressibility, aeration and hydro-elasticity, as revealed by recent experimental studies (e.g. Miyamoto and Tanizawa 1985; Okada and Sumi, 2000; Huera-Huarte *et al.* 2011; Ma *et al.*, 2014, 2015; Mia *et al.*, 2015).

Since Wagner (1932), attempts on deriving analytical solutions or empirical formula have been done to predict the slamming forces, e.g.

Dobrovol'skaya (1969), Armand and Cointe (1986), Cointe (1991). Nevertheless, these analytical works were limited to simple-geometry or wedge-type bodies. In fact, the body shapes and impact angles play important roles on the impact pressure development and fluid-structure formation near the impact surface, as confirmed by the experimental observations, e.g. Okada and Sumi (2000) and Huera-Huarte *et al.* (2011). This limits the extension of the above-mentioned analytical works to bodies with more complex geometry, and therefore, initiated a fast growth of the numerical simulations on the water entry problems.

Many researchers have simulated the water entry problems by using the potential-flow based methods (e.g., Greenhow, 1987; Zhao and Faltinsen, 1993; Zhao *et al.*, 1996). Theoretically, three issues associated with the potential models may limit their applications. The first one is the viscosity of the fluid, which is ignored in the potential models but may be important, particularly on the formation of the local waves around the body surfaces. The second issue is the difficulty on tracking breaking wave surfaces in the potential models, which terminates their modeling when a breaking wave occurs due to their single-phase algorithm. Considering the fact that the maximum load/pressure occurs in a short duration following the entry of the body (often before the formation of broken wave surface), the viscous/turbulent effects may not play insignificant role in terms of the maximum load/pressure on the body and thus the relevant prediction using the potential theory may be acceptable for the cases where the third issue, i.e. the effect of the air (that is ignored in the potential theory), is not important. This issue includes the problems associated with the aeration (Ma *et al.*, 2014) and the compressibility of the fluids (Ma *et al.*, 2015). A recent experimental and numerical work by Lind *et al.* (2015) focusing on the horizontal plate impact on a wave crest or a flat water surface, where the air and water are treated as compressible and incompressible fluids respectively in their multiple-phase Smooth Particle Hydrodynamics (SPH) model based on the solutions of multiple-phase Navier-Stokes (NS) equations, has confirmed the important role of the air phase on modelling the slamming pressure/forces. A similar conclusion has been drawn following earlier experimental studies, e.g. Miyamoto and Tanizawa (1985), who concluded that the impact pressure is reduced and the water surface is deformed before body contacting the water surface. From this point of view, not only the above-mentioned potential models, but also other

single-phase numerical models adopting the NS equation, e.g. Gao *et al.* (2012), Oger *et al.* (2007), Skillen *et al.* (2013) and Zhou & Ma (2010), may not be sufficient for the wave impact problems and a multiple-phase computational fluid dynamics (CFD) models may be necessary.

Such multiple-phase models are usually solved by two types of numerical methods, i.e. the meshless methods and the mesh based methods. In the former, both the fluids and the solid body are represented by Lagrangian particles and the compressibility of the fluids can be considered by involving the equation of the state, e.g. Lind *et al.* (2015). In the latter, the computational domain is discretized into elements/cells and the free surface is identified or tracked by specific techniques, e.g. the volume of fluid (VOF, Kleefsman *et al.* 2005) or level-set (Sussman *et al.*, 1994; Sanders *et al.*, 2010). By using the mesh based methods, the necessity of considering the air phase in the simulation has again been confirmed (e.g. Kleefsman *et al.* 2005; Yang and Qiu, 2012); and, the importance of the compressibility of the air has also been studied (e.g. Yang and Qiu, 2012; Southall *et al.* 2014), leading to a conclusion that the introduction of air compressibility seems to cause a slightly larger peak pressure with lagging of the instant when the peak pressure occurred (Southall *et al.* 2014).

In addition to the free surface identification, another important issue associated with the mesh based methods is the treatment of the computational mesh/grid in order to conform to the moving boundaries of the dropping body. A direct way to solve this problem is the implement of the body conforming moving grid, which only cover the space occupied by the fluids and the mesh is moved following the motion of the body. Two major methods to realize the moving mesh, including the overset grid (e.g. Swidan *et al.*, 2013) and the dynamic mesh method (e.g. Shen and Wan, 2011), are commonly used. Although the body surface boundary conditions can be exactly satisfied and the geometry of the body can be well reserved in these methods, the movement of the mesh may lead to a mesh distortion in the dynamic mesh techniques; a higher-order interpolation and efficient mesh re-construction may be necessary in the overset grid techniques. Alternatively, the embedded fixed grids, where the governing equations are usually discretized on fixed Cartesian grids, are also widely used. By using these grids, the positions and thus the boundaries of the body are identified/approximated by specific techniques, including the volume fraction (Kishev *et al.*, 2006), the cut cell methods (e.g. Yang *et al.*, 1997) and the immersed boundary methods (Zhang *et al.*, 2010; McIntyne *et al.*, 2011; Yang and Stern, 2015; Hesch *et al.*, 2014; Liu and Hu, 2014; Calderer *et al.*, 2014). This way overcomes the mesh distortion caused by the motion of the nodes in the conforming moving grid and has shown a great potential to deal with problems with large motion of the body.

In this paper, comparative numerical studies are carried out. Two different numerical approaches have been used. These include (1) the VOF based multiple-phase solver adopting the dynamic mesh technique available in the OpenFOAM and (2) an in-house multiple-phase immersed boundary method developed recently (Yang, 2015). The results are compared with the experimental data from the third iteration of the Wave Induced Loads on Ships (WILS) Joint Industry Project (MOERI 2013). This also contributes to the comparative study suggested by the International Hydrodynamic Committee, ISOPE, in which the numerical results obtained by different numerical models are compared in terms of the convergence and the accuracy.

## NUMERICAL APPROACHES AND EXPERIMENTAL SETUP

### Numerical Approaches

The first numerical model adopted is the unsteady two-phase solver in OpenFOAM (Jasak *et al.* 2007). The air and water phase are modelled by the ‘one-fluid’ formulation (Tryggvason, Scardovelli, and Zaleski 2011) and the interface is captured by the VOF technique (Kleefsman *et al.* 2005). The rigid body is governed by the 6-DOF model using the Newton’s law. The interaction between the fluid and structure is considered by explicitly imposing the kinematic boundary condition and dynamic boundary condition on the FSI interfaces. A dynamic mesh technology with the Arbitrary-Lagrangian-Eulerian (ALE) (Donea, Giuliani, and Halleux 1982) formulations are implemented. The solver implements the robust transient PIMPLE (merged ISO-SIMPLE) algorithm for the pressure-velocity coupling and a self-adaptive time step is applied to satisfy the Courant–Friedrichs–Lewy (CFL) condition. In order to investigate the role of compressibility of the fluids in the water impact problems, both compressible and incompressible solver may be chosen. The second numerical model is the Immersed boundary method dealing with three-phases (air water and rigid body) interaction, followed the work by Yang (2015). The air and water phases are assumed to be incompressible. Not like most of available immersed boundary method indicated above, the rigid body in Yang (2015) is modelled as a phase of fluid and solved by a generalized equation similar to those for air and water. As a result, there is no explicit treatment of the kinematic and dynamic conditions on the body surface and no need of solving the body motion separately using the Newton’s law. The spatial discretization is based on the standard Marker-and-Cell method in conjunction with a fractional step approach for the pressure/velocity decoupling. In the simulation, the sub-iteration is required to resolve the geometric nonlinearity.

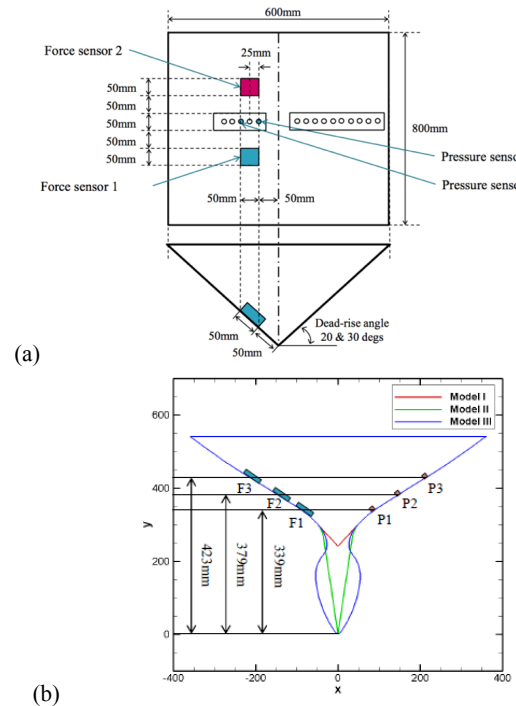


Fig. 1. Schematic description of the model & sensor arrangement for the WILS JIP-III (a: wedge; b: ship-section, MOERI, 2013)

### Experimental Setup

Systematic series of wedge drop tests have been performed in WILS JIP-III. In this paper, however, only limited case with the wedge model

(30-degree deadrise angle, 0 and 20 degree tilting angle) and ship sections are considered due to the limit of the space, as summarised by Table 1. Other results will be presented in the conference. Fig. 1 show a schematic description of the dropping wedge (a) and the ship section (b). More details of the experimental setup and those with ship sections can be found in MOERI (2013) and Kim *et al* (2013).

Table 1. Summary of the test cases

Case No	Model	Drop height (mm)
Wedge-01	Wedge tilting angle 0°	500
Wedge-05	Wedge tilting angle 20°	500
SS-09	Ship Section Model III	170
SS-11	Ship Section Model III	300

Note: the dead-rise angle of the wedge model is 30°

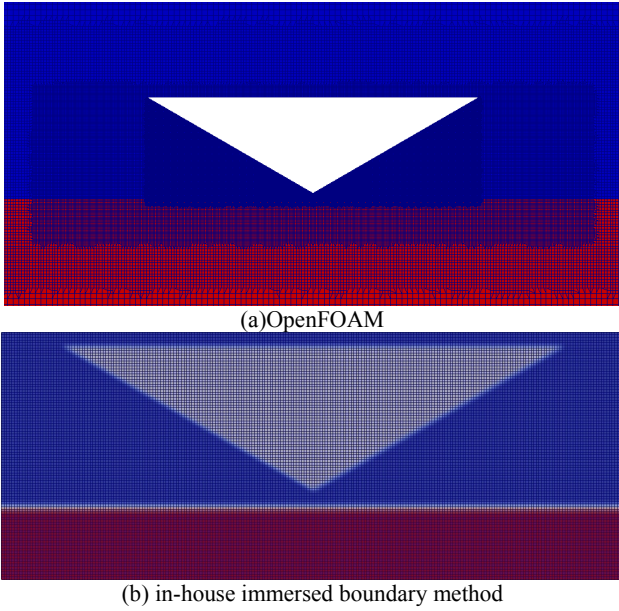


Fig.2 illustration of the computational mesh near the wedge (deadrise angle 30°, tilting angle 0°; color represents volume fraction (red: water; blue: air; white: wedge))

## Numerical Configurations

In the numerical simulation, the densities of the air, water and the wedge are taken as 1.0kg/m<sup>3</sup>, 1000 kg/m<sup>3</sup> and 1643×10<sup>3</sup> kg/m<sup>3</sup>, respectively. The viscosities of the air and the water are assigned to be 10<sup>-5</sup> Pa·s and 10<sup>-3</sup> Pa·s, respectively. Considering the fact that the problems investigated in the experiment is uniform in the longitudinal direction, two-dimensional simulations are carried out by using both the OpenFOAM and the immersed boundary method.

The computational mesh used in the OpenFOAM is hexahedral and a local refinement is adopted in the area where the free surface and violent wave-structure impact may occur. Examples are shown in Fig.2(a) and Fig.3(a) for the cases with the wedge and ship section, respectively. The in-house immersed boundary method adopts a fixed uniform Cartesian grid, as illustrated in Fig.2 (b) and Fig.3(b), in which the solid wedge or ship section is resolved by a level-set algorithm, similar to that used to identify the free surface. Different mesh sizes are used in the convergence tests. A self-adaptive time step size is used in the OpenFOAM simulation and a constant time step size is adopted in the immersed boundary method. For both models, the CFL condition

with the Courant number <0.5, is satisfied.

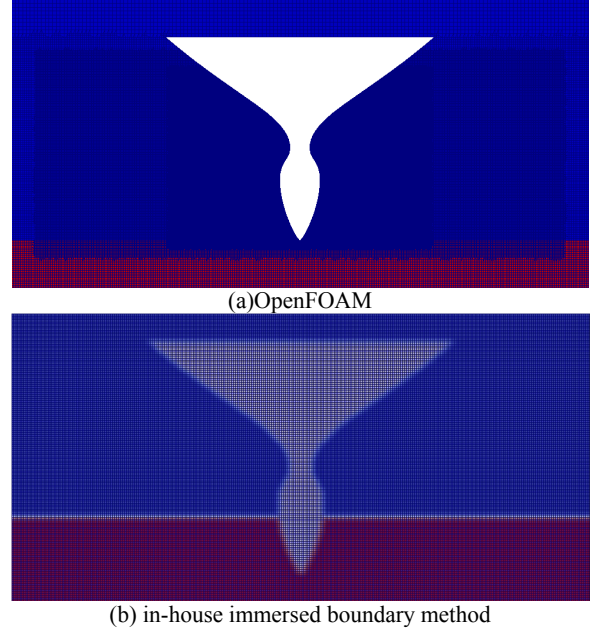


Fig.3 illustration of the computational mesh near the ship section (Model III; color represents volume fraction (red: water; blue: air; white: wedge))

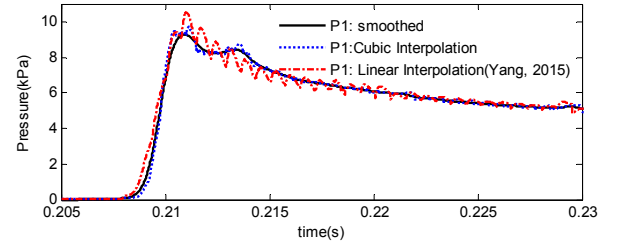


Fig.4 Time histories of pressure recorded by Sensor 1 in the cases with different interpolation schemes (IBM;  $ds = 1\text{mm}$ , Case SS-09)

It shall be noted that considerable fluctuations of the pressure and forces are observed in the IBM simulation that follows Yang (2015), as demonstrated in Fig.4 (dashed-dot line), which illustrates the pressure recorded by Sensor 1 on the dropping ship section in Case SS-09. Such fluctuations are mainly caused by an insufficient accuracy on the level set scheme adopted to realise the dropping body surface. This is evidenced by a considerable improvement in the case with a cubic interpolation scheme (dot line) compared to the original linear interpolation adopted by Yang (2015). Efforts may be devoted to investigating and developing more robust interpolation scheme in order to omit such fluctuations, nevertheless, the present cubic interpolation scheme is used in this comparative study and the time histories of the pressure/forces presented in the rest of the paper are smoothed using 5-point smooth scheme, which seems to reserve the main feature of the pressure/force time histories (solid line).

## CONVERGENCE INVESTIGATION

As indicated above, the Courant condition, i.e. Courant number <0.5, is satisfied in both the OpenFOAM and the immersed boundary method simulations. Therefore, only the convergence properties corresponding to the mesh sizes are considered in the convergence tests. As indicated above, a uniform mesh resolution is adopted in the IBM simulation. In

the cases considered in this paper, the mesh size used by the IBM ranges from 0.5mm to 4mm, yielding corresponding number of cells ranging from 2.1 million to 33k for both the wedge and ship section dropping tests. In the OpenFOAM simulations, the mesh base sizes (maximum mesh size) are the same, i.e. 2cm, but different minimum mesh sizes ( $ds_{min}$ ), which ranging from 0.156mm to 1.25mm, are used using a progressive local refinement. This yields a range of the total number of cells 238k~85k.

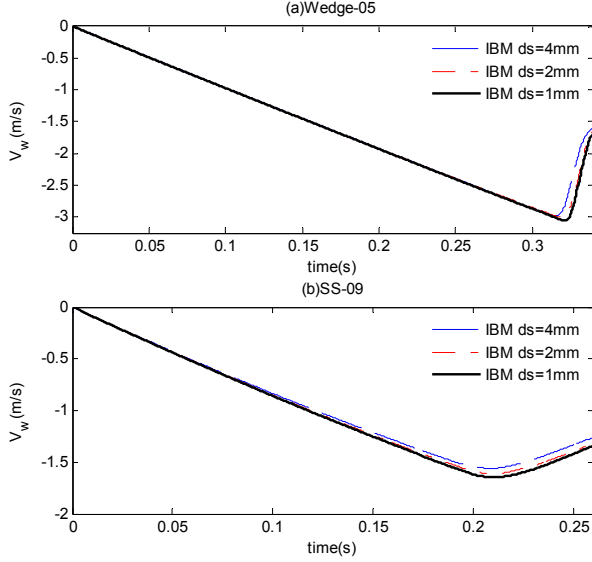


Fig.5 Time histories of velocities of (a) wedge in Wedge-05 and (b) ship section in SS-09 in the cases with different mesh sizes (IBM)

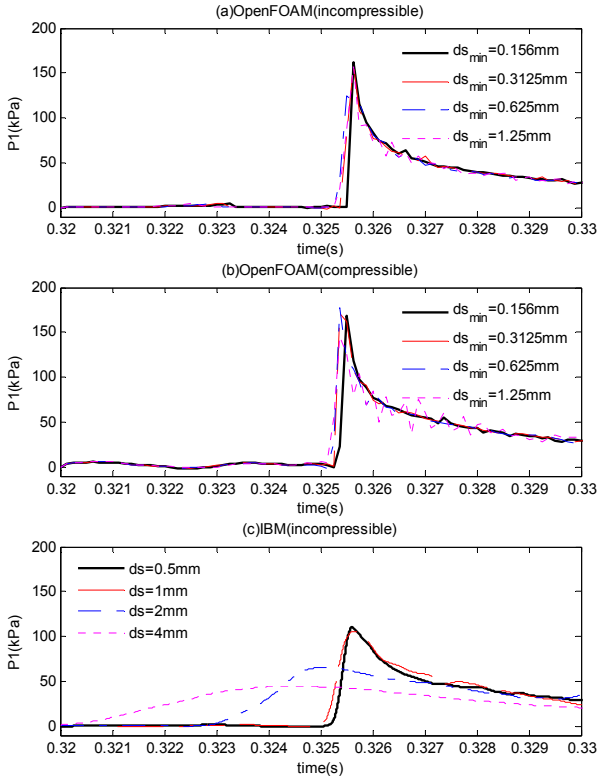


Fig.6 Time histories of pressure by Sensor 1 in the cases with different mesh sizes(Case Wedge-05)

For IBM simulations, it is crucial to examine the motion of the dropping bodies, which is modelled as a phase of fluid and solved by a generalized equation similar to those for air and water (Yang, 2015). Fig.5 compares the time histories of the velocities of dropping wedge (a) or ship section (b) in the cases with different mesh sizes ranging from 1mm to 4mm. Satisfactory agreements have been found between the results with  $ds = 1\text{mm}$  and those with  $ds = 2\text{mm}$ , suggesting that  $ds = 2\text{mm}$  may be sufficient to achieve convergent results of dropping velocity.

The time histories of the pressure and forces acting on the dropping wedges or ship sections are also examined. Fig. 6 illustrates the predicted pressure recorded by the Sensor 1 of Wedge-05 as in the experiment (MOERI, 2013) in the cases with different mesh sizes. It is observed that the results with different values of  $ds_{min}$  are very close, except those with  $ds_{min} = 1.25\text{mm}$ , in the OpenFOAM simulations (both the compressible solver and the incompressible solver, Figs. 6(a) and (b)). A similar phenomenon has also found in the results by the immersed boundary method shown in Figs. 6(c), which suggest that the results with  $ds = 0.5\text{mm}$  and  $ds = 1\text{mm}$  are very close. To further look at the convergence property, the GCI numbers (Celik et al, 2008) are examined for the peak pressure and the rise time. The corresponding GCI numbers for the immersed boundary simulation are 0.0764% and 0.0654% for the peak pressure and rise time at Sensor 1, respectively. They are 0.1922% and 0.1324% at Sensor 2. Similar convergence properties have also been observed in the time histories of the forces on the dropping wedge in this case as demonstrated in Fig. 7.

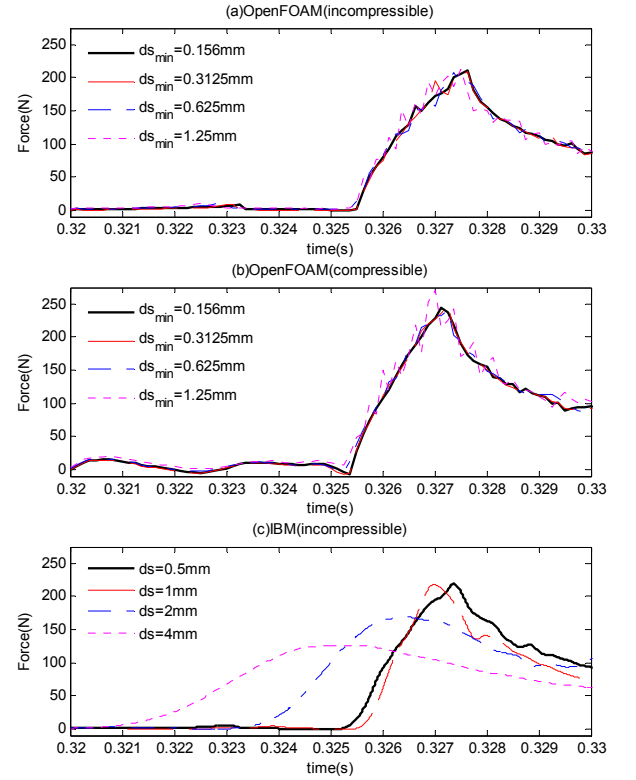


Fig.7 Time histories of force in the cases with different mesh sizes (Case Wedge-05)

It shall be noted that in the convergence investigations shown in Figs. 6 and 7, the numerical simulation is initialized from the state when the wedge velocity reaches  $-3\text{m/s}$  (the distance between the tip of the wedge to the free surface is approximately 1cm) in order to reduce the



size of the computational domain and the CPU time required to complete the simulation. This is justified by the results shown in Fig. 8, which compares the time histories of pressure P1 and force on the wedge in Case Wedge-05, confirming that predicted results in the case modelling the entire procedure (marked as ‘free’) and that initialized from the state when the wedge velocity reaches -3m/s (marked as ‘initialised’) are very close, providing a similar mesh resolution.

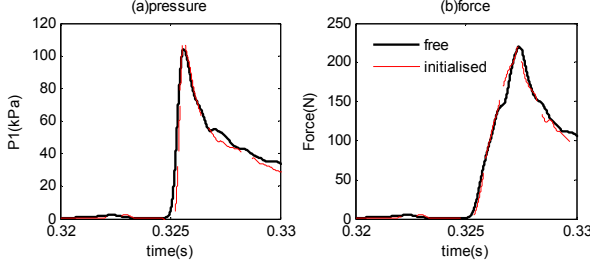


Fig.8 Time histories of pressure and force acting on the wedge in the cases with different case configurations (IBM,  $ds = 0.5\text{mm}$ . Case Wedge-05)

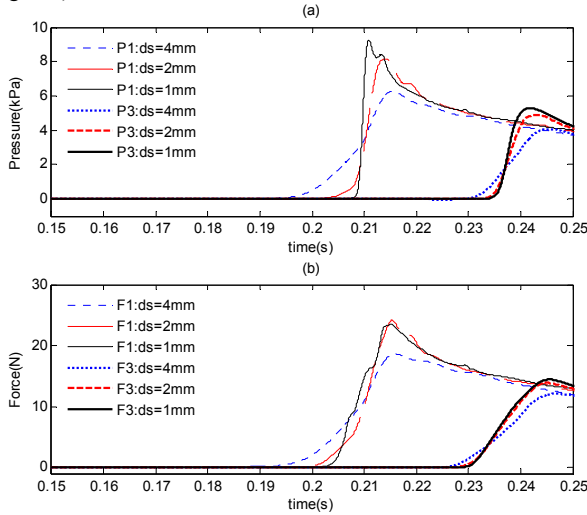


Fig.9 Time histories of (a) pressure and (b) force in the cases with different mesh sizes(IBM; Case SS-09)

Similar convergence tests have also been done for all cases considered in this paper. To save the space, only the corresponding results obtained by the immersed boundary method for Case SS-09, where a ship section model III with 170mm dropping height is considered, are presented in Fig.9 for demonstration. Satisfactory agreements between the results with  $ds = 1\text{mm}$  and those with  $ds = 2\text{mm}$  are observed for P3 and forces recorded at three locations shown in Fig. 1(b). For P1, maximum pressure increases and the rise time decreases as the mesh size decreases from 4mm to 1mm. This is similar to the observation in Southall *et al* (2014), who used OpenFOAM with mesh size ranging from 10mm to 1mm to model the wedge dropping problems.

## COMPARISON WITH EXPERIMENTAL DATA

The numerical results are now compared with the experimental data to shed some light on the accuracy and the reliability of the numerical models on simulating the water entry problems. In this section, the OpenFOAM initialised the simulations of the wedge dropping cases following the approach used in Figs. 6-7, whereas all other numerical simulations follows are initialised as the corresponding experiments.

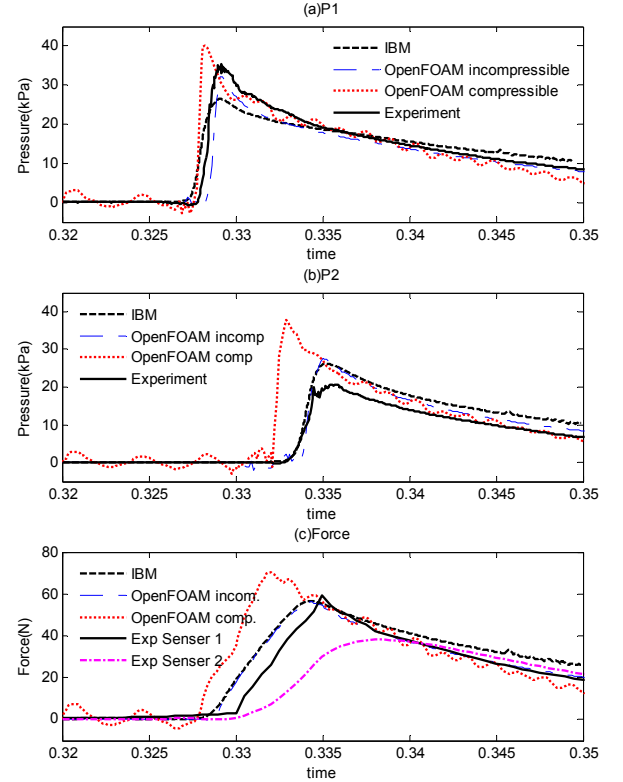


Fig.10 Time histories of pressures and forces on the dropping wedge (Case Wedge-01, OpenFOAM:  $ds_{min} = 0.3125\text{mm}$ ; IBM:  $ds = 1\text{mm}$ )

The first case considered here is Case Wedge-01, in which the wedge with dead-rise angle of  $30^\circ$  and tilting angle of  $20^\circ$  drops from 500mm above the still water surface. Fig.10 compares the time histories of pressures and forces on the dropping wedge in this case. As indicated above, we initialized the OpenFOAM simulations from the state when the wedge velocity reaches -3m/s. Therefore, a time shift of approximately 0.32s has been included in our numerical results. It is noted that in the OpenFOAM compressible solver, only the compressibility of the air has been included and the water phase is assumed to be incompressible. For all pressures and force, the role of the compressibility of the air seems to amplify the peak values and result in a quick rise of the pressure/force. For P1 (Fig.10(a)), the immersed boundary method slightly underestimates the peak pressure; the incompressible OpenFOAM solver results in a result close to the experimental data. For the force, both incompressible OpenFOAM and the immersed boundary method lead to similar results to the experimental data recorded by the force sensor 1, which is quite different from the sensor 2. It is also interesting to observe that the numerical predictions of P2 are considerable larger than the experimental measurement. This is consistent with earlier comparison by Southall *et al* (2014) on the same cases. More significant overestimation of the pressures have been observed in Case Wedge-05 as illustrated in Fig. 11. From Fig.11(a,b), it is found that the difference between the results using the incompressible and compressible OpenFOAM becomes less significant, compared with those shown in Fig. 10; the immersed boundary method give results closer to the experimental data than the OpenFOAM, although there are still more than 100% overestimation of the peak pressure by the immersed boundary method. Compared with the pressure, the overestimation of the force in Case Wedge-05 is relatively insignificant (about 50% overestimation of the peak force).

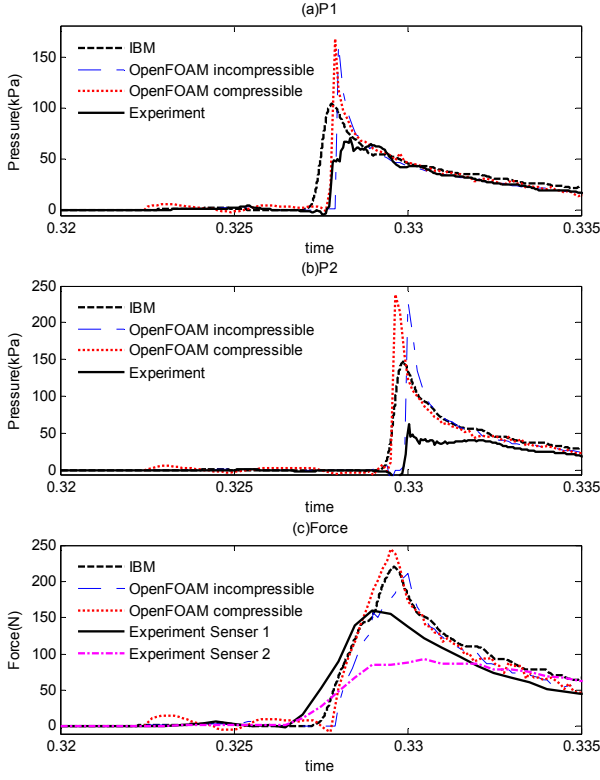


Fig.11 Time histories of pressures and forces on the dropping wedge (Case Wedge-05, dead-rise angle  $30^\circ$ , tilting angle  $20^\circ$ ; OpenFOAM:  $ds_{min}=0.3125\text{mm}$ ; IBM:  $ds=0.5\text{mm}$ )

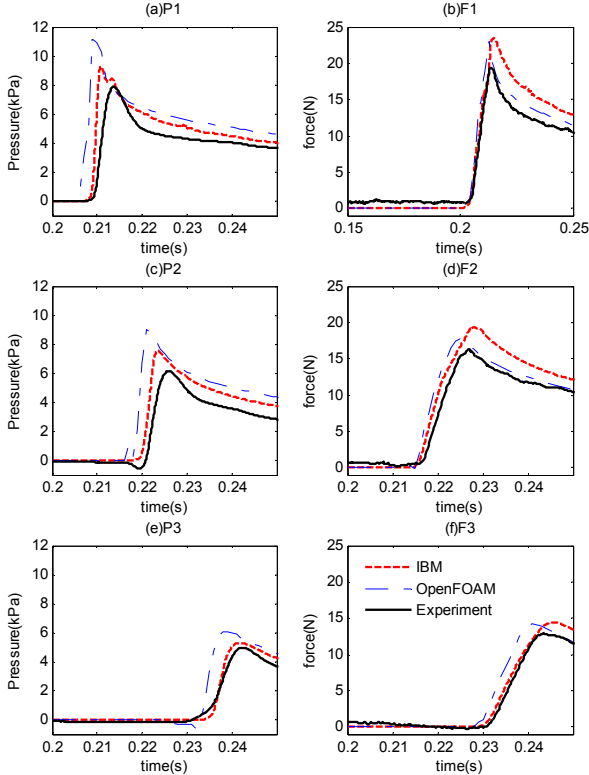


Fig.12 Time histories of pressures and forces on the dropping ship section (Case SS-09, Ship section model III; OpenFOAM: incompressible  $ds_{min}=0.3125\text{mm}$ ; IBM:  $ds=1\text{mm}$ )

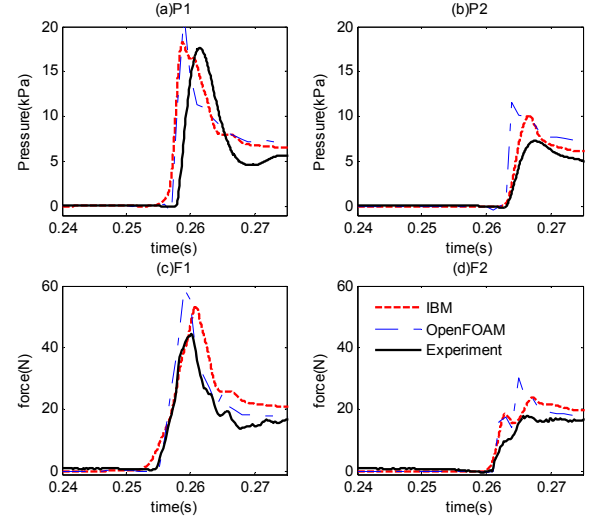


Fig.13 Time histories of pressures and forces on the dropping ship section (Case SS-11, Ship section model III; OpenFOAM: incompressible  $ds_{min}=0.3125\text{mm}$ ; IBM:  $ds=1\text{mm}$ )

Fig.12 compares the time histories of the pressures and forces on the dropping ship section model III in the Case SS-09 with a dropping height of 170mm. It is observed that the incompressible OpenFOAM adopted in this figure overestimate the peak value of both pressure and force; whereas the immersed boundary method is observed to deliver better estimation in terms of the peak value or the rise time. A similar conclusion may also be drawn in Case SS-11, whose results are compared and illustrated in Fig. 13.

In addition to the time histories of the pressure and forces, it is also important to look at the flow fields associated with the water entry problem. Some snapshots of the volume fraction and velocity vectors in different time instants are illustrated in Figs. 14~16 for Case Wedge-01, Case SS-11 and Case SS-09, respectively. For the purpose of comparison, the relevant snapshot in the experimental studies by KRISO are also shown together (the time instant corresponds to the experimental data is roughly estimated from the video file).

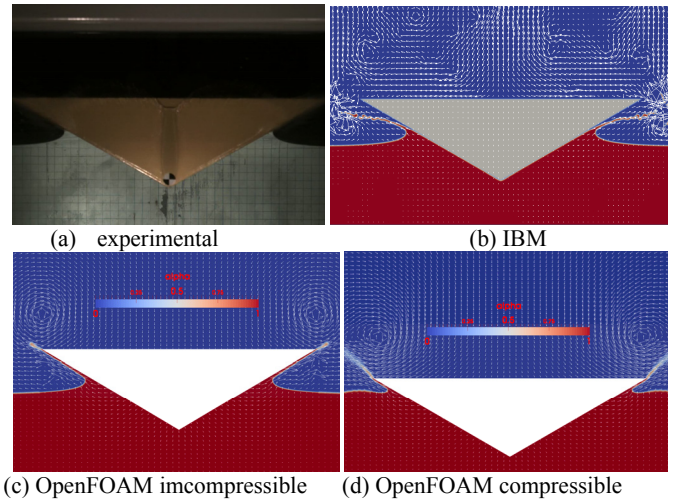


Fig.14 Snapshot of the volume fraction, velocity fields near the dropping wedge at  $t \approx 0.35\text{s}$  (Case Wedge-01; OpenFOAM: incompressible with  $ds_{min}=0.3125\text{mm}$ ; IBM:  $ds=1\text{mm}$ )

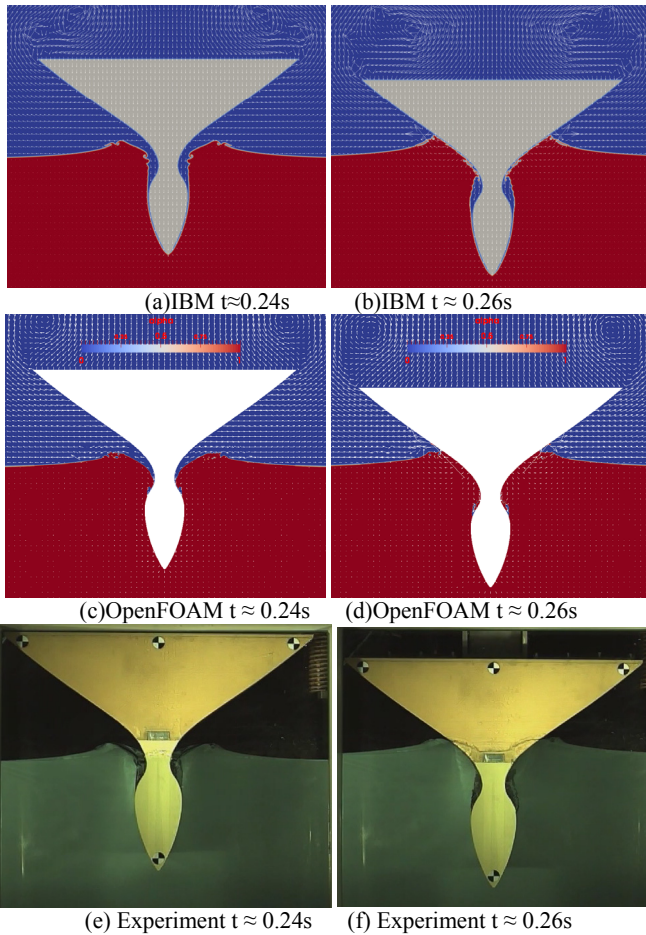


Fig.15 Snapshot of the volume fraction, velocity fields near the dropping ship section (Case SS-11; OpenFOAM: incompressible with  $ds_{min}=0.3125\text{mm}$ ; IBM:  $ds = 1\text{mm}$ )

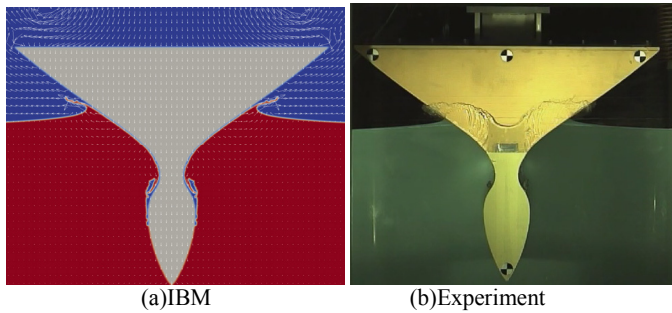


Fig.16 Snapshot of the volume fraction, velocity fields near the dropping ship section at  $t \approx 0.23\text{s}$  (Case SS-09, IBM:  $ds = 1\text{mm}$ )

It is found from Fig. 14 that the position of the wedge predicted by the incompressible OpenFOAM and IBM simulation are close, although different slim jets of the water are observed in these simulations (i.e. attaching to the edge surface in the OpenFOAM and overturning in the IBM simulation). Such slim jets are not observed in the experimental snapshot. Compared with other numerical models, the wedge position predicted by the compressible OpenFOAM is lower, suggesting a quicker dropping. This is consistent with the results shown in Fig. 10 (an earlier rising of the pressure in the case with compressible OpenFOAM compared to other numerical results). For the cases with dropping ship section, an important phenomenon, which is not

observed in the cases with dropping wedge, is that the air may be entrapped as demonstrated by the experimental snapshots in Figs. 15 and 16 for Case SS-11 and SS-09. These are well captured in the IBM simulations as shown in Fig. 15(a&b) and Fig. 16(a). However, in the OpenFOAM simulation, the entrapped air is insignificant (15(d)).

## CONCLUSIONS

In this paper, a comparative study on the water entry problem has been presented. The OpenFOAM and the in-house IBM have been used. Both the wedge and ship section have been considered in the 2D numerical investigation with systematic convergence tests at prior. The results reveal that (1) the IBM can deal with the violent fluid structure interaction problem, e.g. the water entry, without the needs of explicit treatment of the kinematic and dynamic conditions on the body surface; (2) the compressibility of the air plays important role in the impact problem, although the compressible OpenFOAM solver considerably overestimate the pressure and the force; (3) although the compressibility of the fluid is ignored, the IBM seems to be able to well capture the entrapped air during the dropping test; and (4) the reliability of both the OpenFOAM and the immersed boundary is subjected to a satisfactory convergence investigation, since their numerical predictions are sensitive to the selection of the mesh size if insufficient mesh resolution is adopted.

## ACKNOWLEDGEMENTS

The authors gratefully acknowledge the financial support of EPSRC project (EP/N006569/1 and EP/L01467X/1).

## REFERENCES

- Armand, JL, and Cointe, R (1986). "Hydrodynamic Impact Analysis of a Cylinder," *Proceedings of the 15th International Offshore Mechanics and Arctic Engineering*, Vol 1, Tokyo, Japan, pp 609-634.
- Calderer, A, Kang, S and Sotiropoulos, F (2014). Level set immersed boundary method for coupled simulation of air/water interaction with complex floating structures. *Journal of Computational Physics*, 277, 201-227.
- Celik, IB, Ghia, U, and Roache, PJ (2008). "Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications." *Journal of Fluids Engineering-Transactions of the ASME* 130 (7): 078001.
- Cointe, R (1991). "Free Surface Flows Close to a Surface-piercing Body," *Mathematical Approaches in Hydrodynamics*, Society for Industrial and Applied Mathematics, Philadelphia, pp. 319-34.
- Dobrovol'skaya, ZL (1969). "On Some Problems of Similarity Flow of Fluid with a Free Surface," *Journal of Fluid Mechanics*, Vol 36, pp. 805-829.
- Donea, J, Giuliani, S, and J Halleux JP (1982). "An Arbitrary Lagrangian-Eulerian Finite Element Method for Transient Dynamic Fluid-Structure Interactions." *Computer Methods in Applied Mechanics and Engineering*. 33 (1-3). North-Holland: 689-723.
- Gao, R, Ren, B, Wang, G, and Wang, Y (2012). "Numerical Modelling of Regular Wave Slamming on Subface of Open-piled Structures with the Corrected SPH Method," *Applied Ocean Research*, Vol 34, pp 173-186.
- Greenhow, M (1987). "Wedge Entry into Initially Calm Water," *Applied Ocean Research*, Vol 9, pp 214-223.
- Hesch, C, Gil, AJ, Carreño, AA, Bonet, J, and Betsch, P (2014). "A Mortar Approach for Fluid-Structure Interaction Problems: Immersed Strategies for Deformable and Rigid Bodies." *Computer Methods in Applied Mechanics and Engineering*. 278. Elsevier B.V.: 853-82.
- Huerta-Huarte, FJ, Jeon, D, and Gharib, M (2011). "Experimental



- Investigation of Water Slamming Loads on Panels," *Ocean Engineering*, Vol 38, pp 1347-1355.
- Jasak, H, A Jemcov, and Tukovic Z (2007). "OpenFOAM: a C++ Library for Complex Physics Simulations." International Workshop on Coupled Methods in Numerical Dynamics.
- Kim, KH, Hong, SY, Kim, YS, Kim, BW, and Nam, BW (2013). "Experimental Investigation on the Measurement of Impact Force Acting on Two-Dimensional Wedge." In: Seoul.
- Kishev, Z., Hu, C, Kashiwagi, M (2006) "Numerical simulation of violent sloshing by a CIP-based method", *Journal of Marine Science and Technology*, Vol. 11, pp. 111-122.
- Kleefsman, KMT, Fekken G, Veldman, AEP, Iwanowski, B, and Buchner, B (2005). "A Volume-of-Fluid Based Simulation Method for Wave Impact Problems." *Journal of Computational Physics* 206 (1): 363–93.
- Lind, S, Stansby, PK, Rogers, BD, and Lloyd, PM (2015). "Numerical Predictions of Water-air Wave Slam Using Incompressible-smoothed Particle Hydrodynamics," *Applied Ocean Research*, Vol 49, pp 57-71.
- Liu, C. and Hu, C (2014). An efficient immersed boundary treatment for complex moving object. *Journal of Computational Physics*, 274,
- Ma, QW, and Yan, S (2006). "Quasi ALE Finite Element Method for Nonlinear Water Waves." *Journal of Computational Physics* 212 (1): 52–72.
- Ma ZH, Causon DM, Qian L, Mingham CG, Gu HB, Ferrer PM.(2014) "A compressible multiphase flow model for violent aerated wave impact problem." *Proceedings of the Royal Society A*, 470: 20140542.
- Ma, ZH, Qian, L, Causon, DM, Mingham, CG, Mai, T, Greaves, D, and Raby, A. (2015). "The role of fluid compressibility in predicting slamming loads during water entry of flat plates." *Proceedings of the Twenty-fifth (2015) International Ocean and Polar Engineering Conference*, Hawaii, USA
- Mai, T, Hu, ZZ, Greaves, D, and Raby, A (2015). "Investigation of Hydro elasticity: Wave Impact on a Truncated Vertical Wall," *Proceedings of the Twenty-fifth (2015) International Ocean and Polar Engineering Conference*, Hawaii, USA
- McIntyre, SM, Kinzel, MP, Miller, ST, Paterson, EG, Lindau, JW, Kunz, RF(2011) "The immersed boundary method for water entry simulation", 49<sup>th</sup> AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, Orlando, USA.
- Mittal, R, and Gianluca, I (2005). "Immersed Boundary Methods" *Annual Review of Fluid Mechanics* 37 (1). *Annual Reviews*: 239–61.
- Miyamoto, T, and Tanizawa, K (1985). "A Study of the Impact Load on Ship Bow," *Journal of the Society of Naval Architects of Japan*, Vol 60, pp 270-279.
- MOERI (2013). "Wave Induced Loads on Ships." MOERI Technical Report No. BSPIS7230-10306-6.
- Oger, G, Doring, M, Alessandrini, B, and Ferrant, P (2007). "An Improved SPH Method: towards Higher Order Convergence," *Journal of Computational Physics*, Vol 225, pp 1472-1492.
- Okada, S, and Sumi, Y (2000). "On the Water Impact and Elastic Response of a Flat Plate at Small Impact Angles," *Journal of Marine Science and Technology*, Vol 5, pp 31-39.
- Sanders, J, Dolbow, JE, Mucha, PJ, and Laursen, TA (2011). A new method for simulating rigid body motion in incompressible two-phase flow. *International Journal for Numerical Methods in Fluids*, 67(6), 713-732.
- Shen, L, and Chan, ES. (2008). Numerical simulation of fluid–structure interaction using a combined volume of fluid and immersed boundary method. *Ocean Engineering*, 35(8), 939-952.
- Shen, Z, Wan, DC (2011)"Numerical simulation of sphere water entry problem based on VOF and dynamic mesh methods", *Proceedings of the 21<sup>st</sup> International Offshore and Polar Engineering Conference*, Maui, USA
- Skillen, A, Lind, S, Stansby, PK, and Rogers, BD (2013). "Incompressible Smoothed Particle Hydrodynamics (SPH) with Reduced Temporal Noise and Generalized Fickian Smoothing Applied to Body-water Slam and Efficient Wave-body Interaction," *Computer Methods in Applied Mechanics and Engineering*, Vol 265, pp 163-173.
- Southal, NR, Lee, Y, Johnson, MC, Hirdaris, SE, and White, NJ (2014). "Towards a Pragmatic Method for Prediction of Whipping: Wedge Impact Simulations Using OpenFOAM," *Proceedings of the 24th International Offshore and Polar Engineering Conference*, Busan, ISOPE, pp 806-615.
- Sussman, M, Smereka, P, and Osher, S (1994). "A Level Set Approach for Computing Solutions to Incompressible Two-Phase Flow." *Journal of Computational Physics* 114 (1): 146–59.
- Swidan, A, Amin, W, Ranmuthugala, D, Thomas, G, Penesis, I (2013) "Numerical Prediction of Symmetric Water Impact Loads on Wedge Shaped Hull Form Using CFD", *World Journal of Mechanics*, Vol. 3, pp. 311-318
- Thomas, PD, and Lombard CK (2012). "Geometric Conservation Law and Its Application to Flow Computations on Moving Grids." *AIAA Journal* 17 (10): 1030–37.
- Tryggvason, G, Scardovelli, R, and Zaleski, S (2011). "Direct Numerical Simulations of Gas–Liquid Multiphase Flows". Cambridge University Press.
- Wagner, H (1932). "Über stoss und Gleitvorgänge an der Oberfläche von Flüssigkeiten," *Zeitschr. für Angewandte Mathematik und Mechanik*, Vol 12, pp 192-235.
- Yan, S, and Ma, QW (2007). "Numerical Simulation of Fully Nonlinear Interaction Between Steep Waves and 2D Floating Bodies Using the QALE-FEM Method." *Journal of Computational Physics* 221 (2): 666–92.
- Yang G, Causon, D, Ingram, D, Saunders, R., Batten, P (1997) "A Cartesian cut cell method for compressible flows. Part A. Static body problems." *Aeronautical Journal* 101 (1001), 47-56.
- Yang, L (2015). "An Immersed Computational Framework for Multiphase Fluid-Structure Interaction." (Doctoral dissertation). Swansea University.
- Yang, J, and Stern, F (2015). "A Non-Iterative Direct Forcing Immersed Boundary Method for Strongly-Coupled Fluid–Solid Interactions." *Journal of Computational Physics* 295 (C). Elsevier Inc.: 779–804.
- Yang, Q, and Qiu, W (2012). "Numerical Simulation of Water Impact for 2D and 3D Bodies," *Ocean Engineering*, Vol 43, pp 82-89.
- Zhang, Y, Zou, Q, Greaves D, Reeve, D, Hunt-Raby, A, Graham, D, James, P, Lv, X (2010) "A level set immersed boundary method for water entry and exit", *Commun. Comput. Phys.*, Vol. 8, pp. 265-288.
- Zhao, R, and Faltinsen, O (1993). "Water Entry of Two-dimensional Bodies," *Journal of Fluid Mechanics*, Vol 246, pp 593-612.
- Zhao, R, Faltinsen, O, and Aarsnes, J (1996). "Water Entry of Arbitrary Two-dimensional Sections with and without Flow Separation," *Proceedings of 21st Symposium on Naval Hydrodynamics*, Trondheim, Norway.
- Zhou, JT, Ma, QW (2010) "MLPG method based on rankine source solution for modelling 3D breaking waves." *CMES - Computer Modeling in Engineering and Sciences*, 56(2), 179-210.